Model created in COMSOL Multiphysics 6.4

Tubular Centrifuge

Introduction

A tubular centrifuge is a type of centrifugal separation device that is often used for separation of very fine solid particles from a liquid. It can be run in both a continuous and batch type configurations. The device usually consists of a cylindrical rotating bowl with a large aspect ratio where its length is often many times larger than the radius. The bowl has an inlet feed and an outlet.

The particle–liquid mixture enters the rotating bowl through the inlet feed. The large rotation speed of the bowl induces centrifugal forces on the particles which cause them to sediment near the inner walls of the bowl as long as the particle has a larger density compared to the fluid. The sedimentation rate depends on the particle densities and the rotation speeds. Larger rotation speeds enhance the sedimentation rate. Further, the particles with larger densities tend to sediment faster. This dependence on the particle densities can lead to a preferential sedimentation along the axial direction with the most dense particles sedimenting close to the inlet, while the less dense particles sediment at a larger axial distance from the inlet.

This model also demonstrates the process of restarting particle tracing simulations whereby the information of the particles from one study step is used as the initial conditions for a subsequent study step.

Model Definition

The geometry consists of a cylinder of radius 20 mm and a height of 225 mm. There is a circular inlet of radius 5 mm on end of the cylinder through which the particle-fluid mixture enters the cylinder. The mixture exits the cylinder through a circular outlet of radius 5 mm on the face opposite to the inlet. The geometry as presented in Figure 1 shows the cylinder and the outlet.

The particle-fluid mixture sample used in this model is water containing spherical particles of a fixed diameter of 20 μm. The sample contains three types of particles with densities of 1200, 1500, and 2200 kg/m^3. The sample enters the inlet at a flow rate of 2.5 l/min. The fluid entering the cylinder faces a sudden expansion and the corresponding Reynolds number suggests that the flow is turbulent. This model solves the Reynolds-Averaged Navier–Stokes equations and uses the standard k-ω turbulence model to account for the turbulent fluid flow.

The cylinder rotates with an angular speed of 1000 rad/s. The effect of rotation on the fluid flow is modeled using a **Rotating Domain** feature. It is assumed that fluid velocities are stationary and thus the **Frozen Rotor with Initialization** study step is utilized to account for the rotational effects via the inclusion centrifugal and Coriolis forces. Similarly, the rotational effects on the particle motion are included via the **Rotating Frame** feature which adds these fictitious forces to the particles. The **Gravity** feature is used to apply gravitational force in the $-z$ direction. Drag forces are applied in entire domain using the **Drag Force** feature. Inertial effects due to the virtual mass and pressure gradient contributions are also included in the drag force calculations. A **Time Dependent** study step is used to solve the particle trajectories for a total simulation time of 2 s.

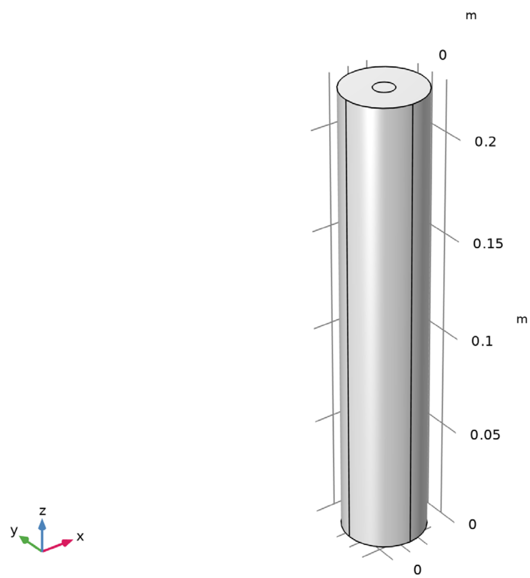

Figure 1: Model geometry of a tubular centrifuge device.

Notes on COMSOL Implementation

The model is solved in two steps. First, the fluid velocity and pressure are solved using the Turbulent Flow, k-ω interface and **Frozen Rotor with Initialization** study step. Then the particle trajectories are solved for using the Particle Tracing for Fluid Flow interface and **Time Dependent** study step.

A total of 3000 particles are released having a fixed diameter of 20 μm. The particle densities are equally distributed over 1200, 1500 and 2200 kg/m^3. The particles are released along the inlet boundary with number density proportional to the fluid velocity along the axial direction. The **Freeze** wall condition is applied at the all the remaining boundaries to represent the sedimentation of the particles on the cylinder walls.

The rotational effects on the fluid and the particles can be accounted for by solving the equations of motion in either the lab-fixed (inertial) frame of reference or a (noninertial) reference frame attached to the cylinder. The analysis in the inertial reference frame would include solving the appropriate equations along with a moving mesh, which can be achieved via the **Rotating Domain** feature. However, the moving mesh computations can be computationally expensive in 3D, especially at large rotational speeds such as those of interest in this example. This can be avoided in situations where the geometry is rotationally invariant about the rotation axis, and the fluid flow is expected to be stationary in the spatial frame. In such situations, the model analysis is best suited in the noninertial reference frame, whereby the rotational effects are included via the addition of fictitious forces such as the centrifugal and the Coriolis forces.

For the fluid flow, this is easily achieved using the **Frozen Rotor with Initialization** study step. For the particle motion, the Particle Tracing Module contains the **Rotating Frame** feature that transforms the coordinate system from the lab-fixed inertial frame to the rotating noninertial frame and includes the fictitious force contributions to the forces on the particles. Since the coordinate system is now attached to a rotating frame, the mesh movement has to be disabled. This is achieved by disabling the **Rotating Domain** feature via the **Modify model configuration for study step** option in the **Time Dependent** study step used to compute the particle trajectories.

The particle trajectories are computed for a total simulation time of 2 seconds. This is split into two **Time Dependent** study steps, with the particle data from the final time step of the first study step being used as the initial conditions for the second study step. In order to accomplish this, it is essential that the **Store particle status data** checkbox is selected in the physics interface settings. The solutions from the two studies are then combined into one solution using the **Combine Solutions** study step.

Note that the particle tracing for an overall simulation time of 2 seconds can also be achieved directly in one study step. This is split into two study steps in this model to demonstrate the technique of restarting the particle tracing simulations and combining the solutions from two studies.

Results and Discussion

Figure 2 shows the velocity field (z direction) and the streamlines. The streamlines are colored based on the z-component of the fluid velocity. As the flow is fully developed, the velocity at center of the inlet is higher. The sudden expansion in the geometry causes the fluid velocity in the axial direction to drop immediately after the inlet. The rotational effects are clearly evident in the streamlines.

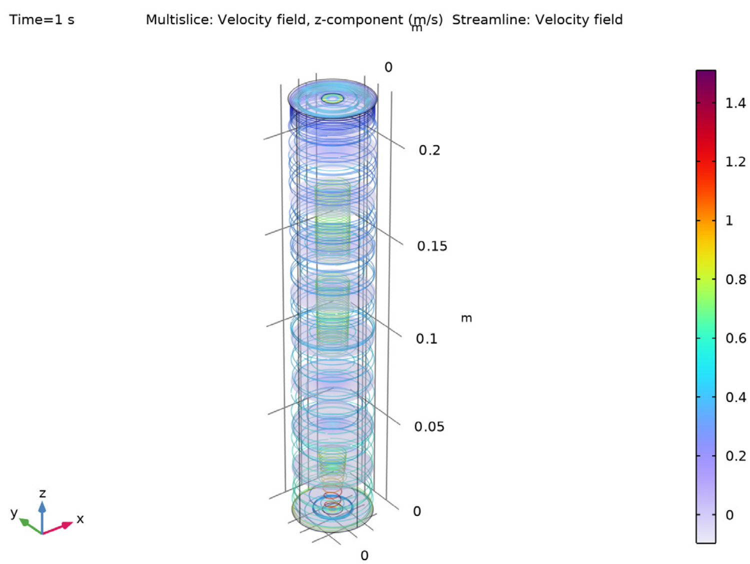

Figure 2: Velocity field and streamlines for the fluid flow.

Figure 3 show the particle trajectories at the end of 2 s. The particles are colored by their densities. The fluid carries the particles toward the outlet, while the centrifugal forces push the particles toward the cylinder walls. Denser particles are carried by the water for shorter distances and end up sedimenting closer to the inlet, while the less dense particles are carried further along. A large portion of the particles with the lowest density are found to sediment close to the outlet.

This preferential sedimentation is visualized in Figure 4, where the distribution of particle densities as a function of their z-coordinate is plotted at the end of 2 s. Finally, Figure 5 shows the number of particles sedimented on the walls as a function of time for each particle type. This illustrates the dependence of the sedimentation rates on the particle density.

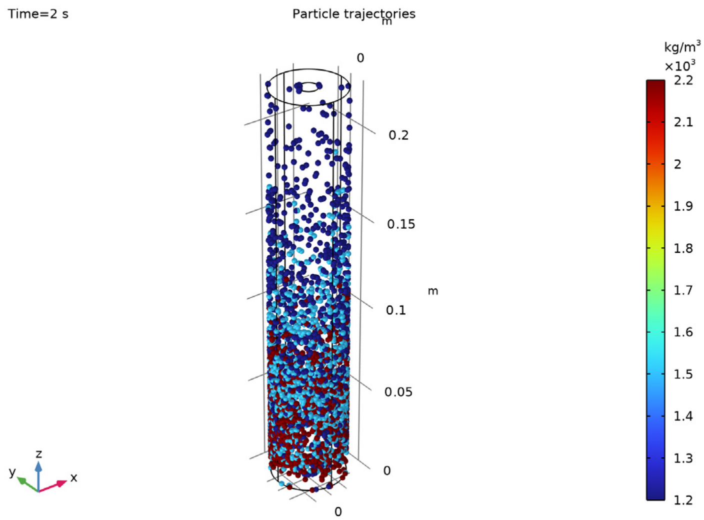

Figure 3: Particle trajectories colored by particle diameter.

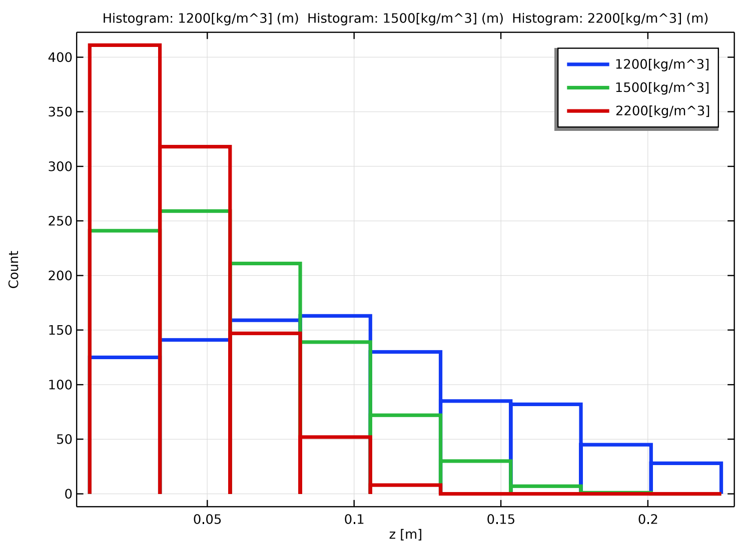

Figure 4: 1D histogram showing the distribution of particle densities collected at the wall as a function of the z-coordinate.

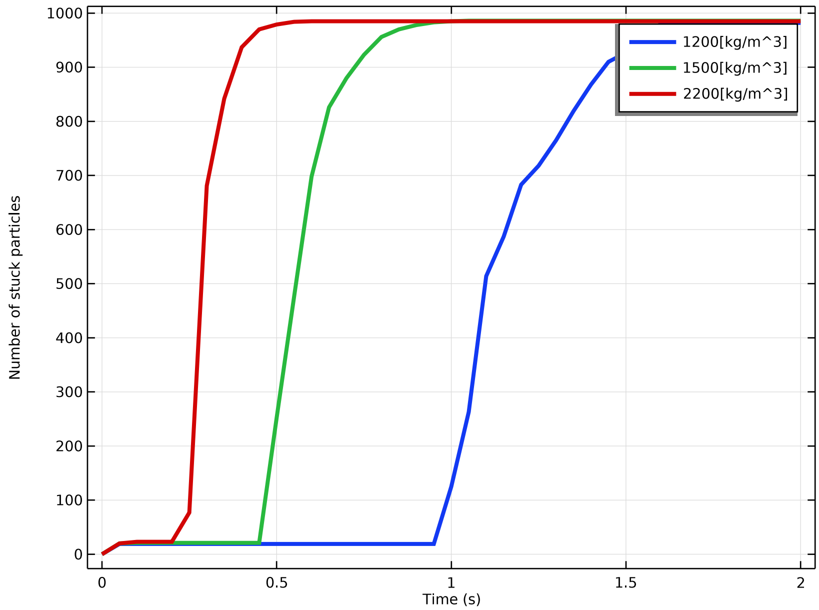

Figure 5: Number of sedimented particles as a function of time.

Application Library path: `Particle_Tracing_Module/Fluid_Flow/tubular_centrifuge`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.
2 In the **Select Physics** tree, select **Fluid Flow** > **Single-Phase Flow** > **Turbulent Flow** > **Turbulent Flow, k-ω** (spf).
3 Click **Add**.

4 In the **Select Physics** tree, select **Mathematics** > **Deformed Mesh** > **Moving Mesh** > **Rotating Domain**.

5 Click **Add**.

6 In the **Select Physics** tree, select **Fluid Flow** > **Particle Tracing** > **Particle Tracing for Fluid Flow (fpt)**.

7 Click **Add**.

8 Click **Study**.

9 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces** > **Frozen Rotor with Initialization**.

10 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
omega	1000[rad/s]	1000 rad/s	Angular speed
L_tube	225[mm]	0.225 m	Tube length
R_tube	20[mm]	0.02 m	Tube radius
R_feed	5[mm]	0.005 m	Feed radius
Vol_f	2.5[l/min]	4.1667E-5 m^3/s	Fluid volumetric flow rate
dia_part	20[um]	2E-5 m	Particle diameter
num_rel	1000	1000	Number of particles per type

GEOMETRY 1

Cylinder 1 (cyl1)

1 In the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `R_tube`.

4 In the **Height** text field, type `L_tube`.

5 Click **Build Selected**.

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work Plane**.

Work Plane 1 (wp1) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1) > Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `R_feed`.

4 Click **Build Selected**.

Work Plane 2 (wp2)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Duplicate**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 In the **z-coordinate** text field, type `L_tube`.

Form Union (fin)

1 In the **Geometry** toolbar, click **Build All**.

2 Click the **Transparency** button in the **Graphics** toolbar.

DEFINITIONS

Create explicit selections for the inlet, outlet and the walls of the cylinder.

Inlet

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Inlet` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 5 only.

Outlet

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Outlet` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 6 only.

Walls

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Walls` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 1–4, 7, and 8 only.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in** > **Water, liquid**.

4 Click the **Add to Component** button in the window toolbar.

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

Now specify the settings for the **Rotating Domain** feature which usually controls the moving mesh. Even though this model solves the governing equations (for both the fluid and the particles) in a noninertial reference frame and thus avoids the need for a moving mesh, this node is needed for the **Frozen Rotor** study step.

MOVING MESH

Rotating Domain 1

1 In the **Settings** window for **Rotating Domain**, locate the **Rotation** section.

2 From the **Rotation type** list, choose **Specified rotational velocity**.

3 In the ω text field, type `omega`.

TURBULENT FLOW, K-ω (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, k-ω (spf)**.

2 In the **Settings** window for **Turbulent Flow, k-ω**, locate the **Physical Model** section.

3 Select the **Include gravity** checkbox.

Inlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Inlet**.

4 Locate the **Boundary Condition** section. From the list, choose **Fully developed flow**.

5 Locate the **Fully Developed Flow** section. Click the **Flow rate** button.

6 In the V_0 text field, type Vol_f.

Outlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 In the **Settings** window for **Outlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Outlet**.

4 Locate the **Pressure Conditions** section. Select the **Normal flow** checkbox.

PARTICLE TRACING FOR FLUID FLOW (FPT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Particle Tracing for Fluid Flow (fpt)**.

2 In the **Settings** window for **Particle Tracing for Fluid Flow**, locate the **Additional Variables** section.

3 Select the **Store particle status data** checkbox. This is necessary to use the particle trajectories from one study step as initial conditions for a subsequent study step.

Particle Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Particle Tracing for Fluid Flow (fpt)** click **Particle Properties 1**.

2 In the **Settings** window for **Particle Properties**, locate the **Particle Properties** section.

3 From the ρ_p list, choose **User defined**. In the associated text field, type 1200[kg/m^3].

4 In the d_p text field, type dia_part.

Particle Properties 2

1 In the **Physics** toolbar, click **Global** and choose **Particle Properties**.

2 In the **Settings** window for **Particle Properties**, locate the **Particle Properties** section.

3 From the ρ_p list, choose **User defined**. In the associated text field, type 1500[kg/m^3].

4 In the d_p text field, type dia_part.

Particle Properties 3

1 Right-click **Particle Properties 2** and choose **Duplicate**.

2 In the **Settings** window for **Particle Properties**, locate the **Particle Properties** section.

3 In the ρ_p text field, type 2200[kg/m^3].

Rotating Frame 1

1 In the **Physics** toolbar, click **Domains** and choose **Rotating Frame**.

2 In the **Settings** window for **Rotating Frame**, locate the **Rotating Frame** section.

3 In the Ω text field, type `omega`.

Drag Force 1

1. In the **Physics** toolbar, click **Domains** and choose **Drag Force**.
2. Select Domain 1 only.
3. In the **Settings** window for **Drag Force**, locate the **Drag Force** section.
4. From the **Drag law** list, choose **Schiller–Naumann**.
5. From the **u** list, choose **Velocity field (spf)**.
6. Locate the **Additional Terms** section. Select the **Include virtual mass and pressure gradient forces** checkbox. This is important to capture the inertial effects of the particle.

Gravity Force 1

1. In the **Physics** toolbar, click **Domains** and choose **Gravity Force**.
2. Select Domain 1 only.

Inlet 1

Release the particles such that number density of the particles is proportional to the axial component of the fluid density at the inlet. The fluid velocity at the inlet is used to initialize the particle velocities. Note that since the fluid velocity is defined in an inertial reference frame and the particle velocities are defined in the rotating (noninertial) reference frame, the moving frame velocity must be subtracted from the fluid velocity.

1. In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.
2. In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Inlet**.
4. Locate the **Initial Position** section. From the **Initial position** list, choose **Density**.
5. In the N text field, type `num_rel`.
6. In the ρ text field, type `w`.
7. Locate the **Initial Velocity** section. From the **u** list, choose **Velocity field (spf)**.
8. Click to expand the **Advanced Settings** section. Select the **Subtract moving frame velocity from initial particle velocity** checkbox.

Inlet 2

1. Right-click **Inlet 1** and choose **Duplicate**.
2. In the **Settings** window for **Inlet**, click to expand the **Released Particle Properties** section.
3. From the **Released particle properties** list, choose **Particle Properties 2**.

Inlet 3

1. Right-click **Inlet 2** and choose **Duplicate**.
2. In the **Settings** window for **Inlet**, locate the **Released Particle Properties** section.
3. From the **Released particle properties** list, choose **Particle Properties 3**.

Outlet 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.
2. In the **Settings** window for **Outlet**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Outlet**.

Particle Counter 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Particle Counter**.
2. In the **Settings** window for **Particle Counter**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Walls**.
4. Locate the **Particle Counter** section. From the **Particle selection** list, choose **Particle properties**.
5. From the **Released particle properties** list, choose **Particle Properties 1**.

Particle Counter 2

1. Right-click **Particle Counter 1** and choose **Duplicate**.
2. In the **Settings** window for **Particle Counter**, locate the **Particle Counter** section.
3. From the **Released particle properties** list, choose **Particle Properties 2**.

Particle Counter 3

1. Right-click **Particle Counter 2** and choose **Duplicate**.
2. In the **Settings** window for **Particle Counter**, locate the **Particle Counter** section.
3. From the **Released particle properties** list, choose **Particle Properties 3**.

Add a **Time Dependent** study step to follow the **Frozen Rotor** study step. This study step only solves for the Particle Tracing for Fluid Flow interface. Further, since the usage of the **Rotating Frame** feature removes the need for a moving mesh, the **Rotating Domain** feature must be disabled for this study step.

STUDY 1

Step 3: Time Dependent

1. In the **Study** toolbar, click **Time Dependent**.
2. In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,0.05,1.0)`.

4 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Moving Mesh** and **Turbulent Flow, k-ω (spf)**.

5 Select the **Modify model configuration for study step** checkbox.

6 In the tree, select **Component 1 (comp1)** > **Moving Mesh, Controls spatial frame** > **Rotating Domain 1**.

7 Click **Disable**.

8 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

9 From the **Method** list, choose **Solution**.

10 From the **Study** list, choose **Study 1, Frozen Rotor**.

11 In the **Study** toolbar, click **Compute**.

RESULTS

Multislice 1

1 In the **Model Builder** window, expand the **Velocity (spf)** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Expression** section.

3 In the **Expression** text field, type `w`.

4 Locate the **Multiplane Data** section. Find the **x-planes** subsection. In the **Planes** text field, type `0`.

5 Find the **y-planes** subsection. In the **Planes** text field, type `0`.

6 Find the **z-planes** subsection. From the **Entry method** list, choose **Coordinates**.

7 In the **Coordinates** text field, type `range(0,(L_tube-0)/9,L_tube)`.

8 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.

Velocity (spf)

In the **Velocity (spf)** toolbar, click **Streamline**.

Streamline 1

1 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

2 From the **Positioning** list, choose **Starting-point controlled**.

3 In the **Points** text field, type `100`.

4 In the **Velocity (spf)** toolbar, click **Plot**.

Color Expression 1

1 In the **Velocity (spf)** toolbar, click **Color Expression**.

2 In the **Settings** window for **Color Expression**, locate the **Expression** section.

3 In the **Expression** text field, type `w`.

4 Locate the **Coloring and Style** section. Clear the **Color legend** checkbox. The plot should look like Figure 2.

Particle Trajectories (fpt)

1 In the **Model Builder** window, expand the **Results** > **Particle Trajectories (fpt)** node, then click **Particle Trajectories (fpt)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.

3 Select the **Show units** checkbox.

Particle Trajectories 1

1 Click the **Transparency** button in the **Graphics** toolbar.

2 In the **Model Builder** window, click **Particle Trajectories 1**.

3 In the **Settings** window for **Particle Trajectories**, locate the **Coloring and Style** section.

4 Find the **Point style** subsection.

5 Select the **Radius scale factor** checkbox. In the associated text field, type `1.5`.

6 In the **Particle Trajectories (fpt)** toolbar, click **Plot**.

Color Expression 1

1 In the **Model Builder** window, expand the **Particle Trajectories 1** node, then click **Color Expression 1**.

2 In the **Settings** window for **Color Expression**, locate the **Expression** section.

3 In the **Expression** text field, type `fpt.rhop`.

4 In the **Particle Trajectories (fpt)** toolbar, click **Plot**.

The particles have not yet been fully separated from the fluid. In order to simulate the particle trajectories for an additional 1 second, it is possible to utilize the current solution as the initial conditions for a subsequent study step.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Time Dependent**.

4 Click the **Add Study** button in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 In the **Output times** text field, type `range(1.0,0.05,2.0)`.

3 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Moving Mesh** and **Turbulent Flow, k-ω (spf)**.

4 Select the **Modify model configuration for study step** checkbox.

5 In the tree, select **Component 1 (comp1) > Moving Mesh, Controls spatial frame > Rotating Domain 1**.

6 Click **Disable**.

7 Locate the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

8 From the **Method** list, choose **Solution**.

9 From the **Study** list, choose **Study 1, Time Dependent**.

10 From the **Time (s)** list, choose **Last**.

11 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

12 From the **Method** list, choose **Solution**.

13 From the **Study** list, choose **Study 1, Time Dependent**.

14 In the **Study** toolbar, click **Compute**.

RESULTS

Particle Trajectories (fpt) 1

Now combine the solutions from the two particle tracing study steps into one solution. The solution at the first timestep in the **Study 2** should be deleted while concatenating the two solutions.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Empty Study**.

4 Click the **Add Study** button in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

Step 1: Combine Solutions

1 In the **Study** toolbar, click **More Study Extensions** and choose **Combine Solutions**.

2 In the **Settings** window for **Combine Solutions**, locate the **Combine Solutions Settings** section.

3 From the **Solution operation** list, choose **Remove solutions**.

4 From the **Solution** list, choose **Study 2/Solution 4 (sol4)**.

5 From the **Time (s)** list, choose **Manual**.

Step 2: Combine Solutions 2

1 In the **Study** toolbar, click **More Study Extensions** and choose **Combine Solutions**.

2 In the **Settings** window for **Combine Solutions**, locate the **Combine Solutions Settings** section.

3 From the **First solution** list, choose **Study 1/Solution 1 (sol1)**.

4 In the **Study** toolbar, click **Compute**.

Once the solutions from the two particle tracing studies are combined, create a new **Particle** dataset from the new solution.

RESULTS

Particle 3

1 In the **Results** toolbar, click **More Datasets** and choose **Particle**.

2 In the **Settings** window for **Particle**, locate the **Particle Solution** section.

3 From the **Solution** list, choose **Solution 5 (sol5)**.

Particle Trajectories (fpt)

1 In the **Model Builder** window, under **Results** click **Particle Trajectories (fpt)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Particle 3**. The plot should look like Figure 3.

Animation 1

1 In the **Results** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Scene** section.

3 From the **Subject** list, choose **Particle Trajectories (fpt)**.

4 Locate the **Frames** section. From the **Frame selection** list, choose **All**.

5 Click the **Play** button in the **Graphics** toolbar.

z histogram

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `z histogram` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Particle 3**.

4 From the **Time selection** list, choose **Last**.

5 Locate the **Plot Settings** section. Select the **x-axis label** checkbox.

6 Select the **y-axis label** checkbox.

7 In the **x-axis label** text field, type `z [m]`.

8 In the **y-axis label** text field, type `Count`.

Histogram 1

1 In the **z histogram** toolbar, click **Histogram**.

2 In the **Settings** window for **Histogram**, locate the **Expression** section.

3 In the **Expression** text field, type `if(fpt.sidx==1,qz,0)`.

4 Select the **Description** checkbox. In the associated text field, type `1200[kg/m^3]`.

5 Locate the **Bins** section. From the **Entry method** list, choose **Limits**.

6 In the **Limits** text field, type `range(0.01,(L_tube-0.01)/9,L_tube)`.

7 Locate the **Output** section. From the **Function** list, choose **Discrete**.

8 Click to expand the **Coloring and Style** section. From the **Width** list, choose **3**.

9 Click to expand the **Legends** section. Select the **Show legends** checkbox.

10 Find the **Include** subsection. Clear the **Solution** checkbox.

11 Select the **Description** checkbox.

12 In the **z histogram** toolbar, click **Plot**.

Histogram 2

1 Right-click **Histogram 1** and choose **Duplicate**.

2 In the **Settings** window for **Histogram**, locate the **Expression** section.

3 In the **Expression** text field, type `if(fpt.sidx==2,qz,0)`.

4 In the **Description** text field, type `1500[kg/m^3]`.

Histogram 3

1 Right-click **Histogram 2** and choose **Duplicate**.

2 In the **Settings** window for **Histogram**, locate the **Expression** section.

3 In the **Expression** text field, type `if(fpt.sidx==3,qz,0)`.

4 In the **Description** text field, type `2200[kg/m^3]`.

z histogram

1 In the **Model Builder** window, click **z histogram**.

2 In the **z histogram** toolbar, click **Plot**. The plot should look like Figure 4.

Particle Counters

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Particle Counters` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Particle 3**.

4 Locate the **Plot Settings** section. Select the **x-axis label** checkbox.

5 Select the **y-axis label** checkbox.

6 In the **x-axis label** text field, type `Time (s)`.

7 In the **y-axis label** text field, type `Number of stuck particles`.

Global 1

1 In the **Particle Counters** toolbar, click **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
fpt.pcnt1.Nsel	1	1200[kg/m^3]
fpt.pcnt2.Nsel	1	1500[kg/m^3]
fpt.pcnt3.Nsel	1	2200[kg/m^3]

4 Click to expand the **Coloring and Style** section. From the **Width** list, choose **3**.

5 Click to expand the **Legends** section. In the **Particle Counters** toolbar, click **Plot**. The plot should look like Figure 5.